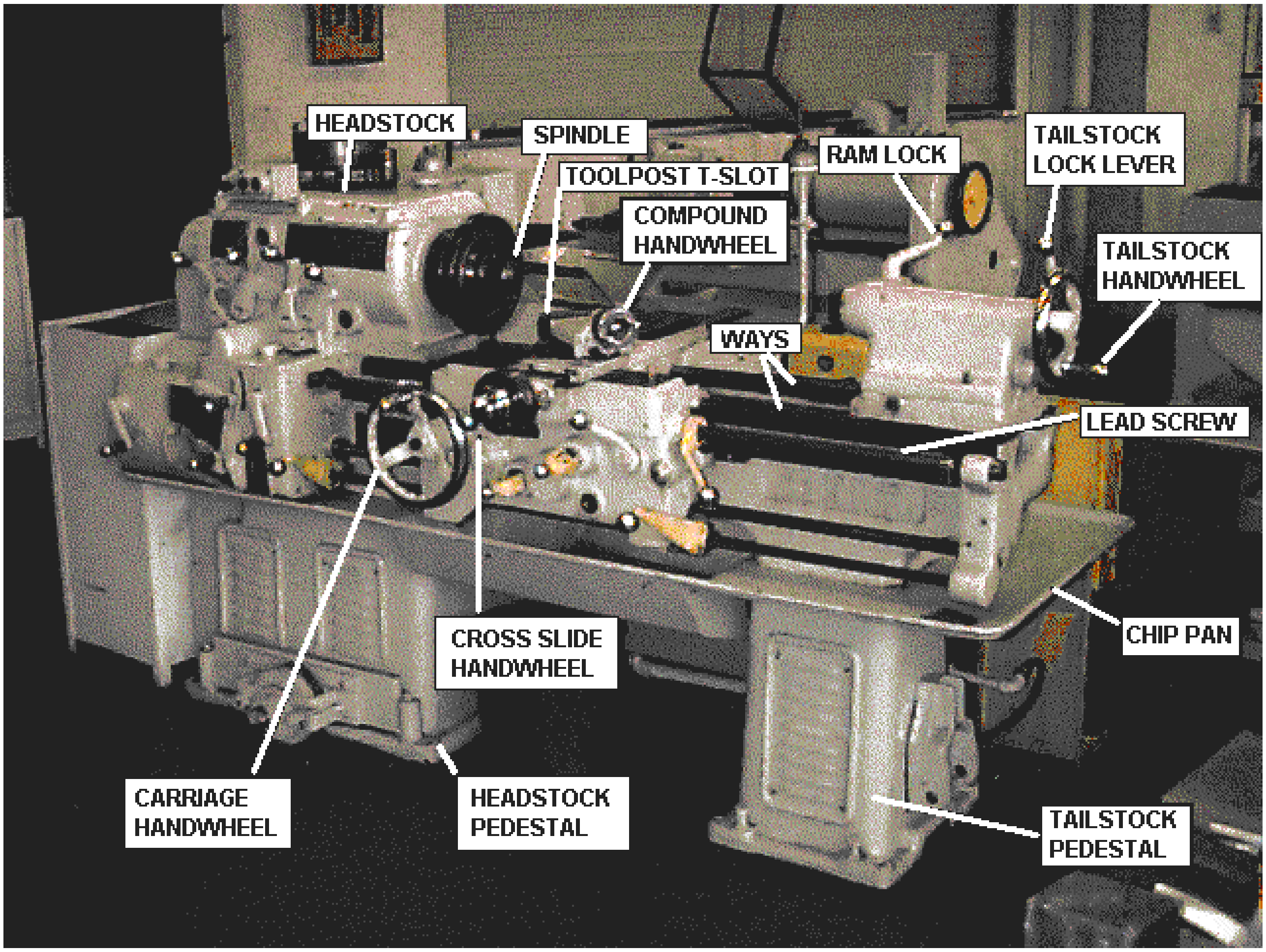
Lathe



A Lathe in Building 35

Description

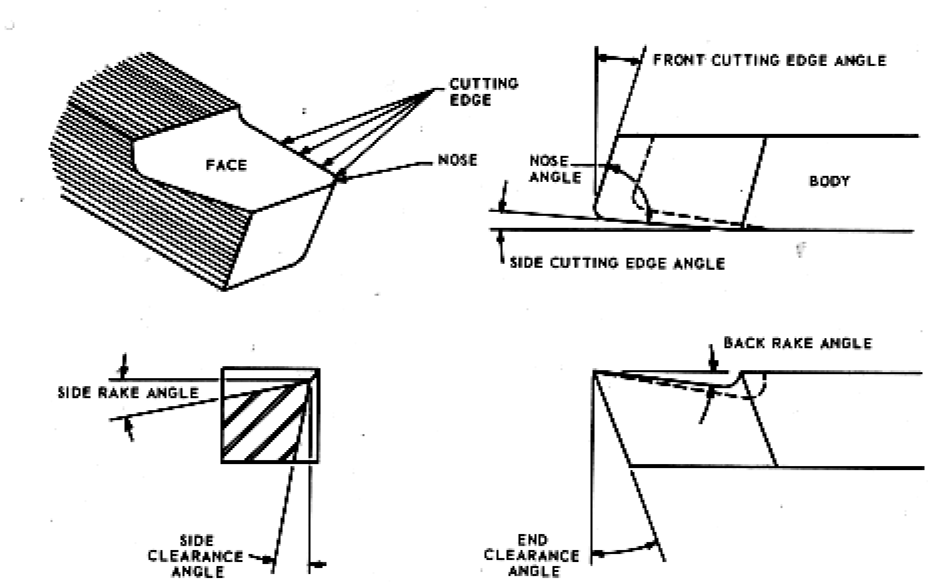
The purpose of a lathe is to rotate a part against a tool whose position it controls. It is useful for fabricating parts and/or features that have a circular cross section. The spindle is the part of the lathe that rotates. Various workholding attachments such as three jaw chucks, collets, and centers can be held in the spindle. The spindle is driven by an electric motor through a system of belt drives and/or gear trains. Spindle speed is contolled by varying the geometry of the drive train.

The tailstock can be used to support the end of the workpiece with a center, or to hold tools for drilling, reaming, threading, or cutting tapers. It can be adjusted in position along the ways to accomodate different length workpices. The ram can be fed along the axis of rotation with the tailstock handwheel.

The carriage controls and supports the cutting tool. It consists of:

- A saddle that mates with and slides along the ways.
- An apron that controls the feed mechanisms.
- A cross slide that controls transverse motion of the tool (toward or away from the operator).
- A tool compound that adjusts to permit angular tool movement.
- A toolpost T-slot that holds the toolpost.

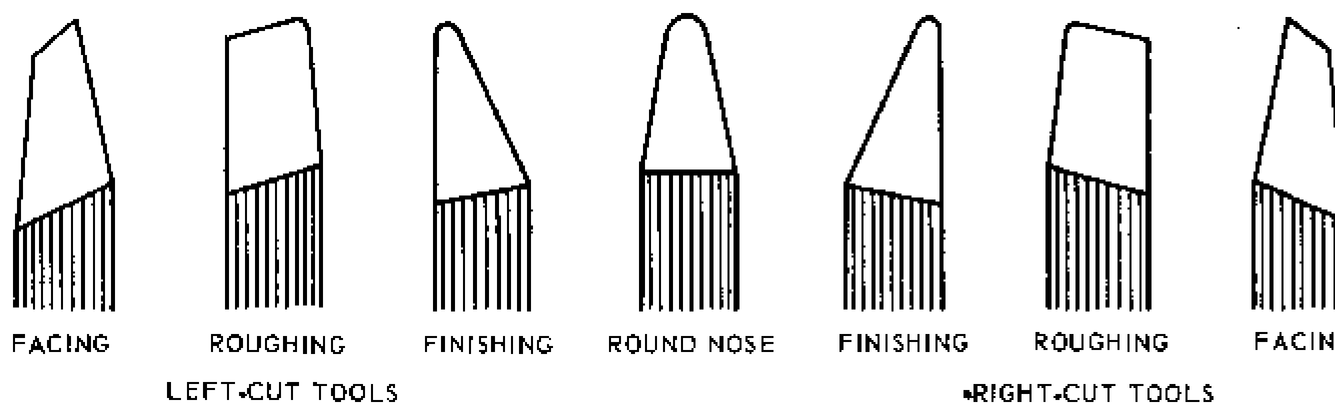
Choosing a Cutting Tool



Cutting Tool Terminology

The figure above shows a typical cutting tool and the terminology used to describe it. The actual geometry varies with the type of work to be done. The standard cutting tool shapes are shown below.

- Facing tools are ground to provide clearance with a center.
- Roughing tools have a small side relief angle to leave more material to support the cutting edge during deep cuts.
- Finishing tools have a more rounded nose to provide a finer finish. Round nose tools are for lighter turning. They have no back or side rake to permit cutting in either didection.
- Left hand cutting tools are designed to cut best when traveling from left to right.
- Aluminum is cut best by specially shaped cutting tools (not shown) that are used with the cutting edge slightly above center to reduce chatter.



Standard Cutting Tools

Installing a Cutting Tool

Lathe cutting tools are held by tool holders. To install a tool, first clean the holder, then tighten the bolts. *(click here for video) 2.11MB*

The tool post is secured to the compound with a T-bolt. The tool holder is secured to the tool post using a quick release lever. *(click here for video) 1.99MB*

Positioning the Tool

In order to move the cutting tool, the lathe saddle and cross slide can be moved by hand. *(click here for video) 860kB*

There are also power feeds for these axes. Procedures vary from machine to machine.

A third axis of motion is provided by the compound. The angle of the compound can be adjusted to allow tapers to be cut at any desired angle. First, loosen the bolts securing the compound to the saddle. Then rotate the compound to the desired angle referencing the dial indicator at the base of the compound. Retighten the bolts. Now the tool can be hand fed along the desired angle. No power feed is available for the compound. If a fine finish is required, use both hands to achieve a smoother feed rate. *(click here for video) 2.05MB*

The cross slide and compound have a micrometer dial to allow accurate positioning, but the saddle doesn't. To position the saddle accurately, you may use a dial indicator mounted to the saddle. The dial indicator presses against a stop (often a micrometer as shown in the clip below). *(click here for video) 1.29MB*

Feed, Speed, and Depth of Cut

Cutting speed is defined as the speed at which the work moves with respect to the tool (usually measured in feet per minute). Feed rate is defined as the distance the tool travels during one revolution of the part. Cutting speed and feed determines the surface finish, power requirements, and material removal rate. The primary factor in choosing feed and speed is the material to be cut. However, one should also consider material of the tool, rigidity of the workpiece, size and condition of the lathe, and depth of cut. For most Aluminum alloys, on a roughing cut (.010 to .020 inches depth of cut) run at 600 fpm. On a finishing cut (.002 to .010 depth of cut) run at 1000 fpm. To calculate the proper spindle speed, divide the desired cutting speed by the circumference of the work. Experiment with feed rates to achieve the desired finish. In considering depth of cut, it's important to remember that for each thousandth depth of cut, the work diameter is reduced by *two* thousandths.

Turning

The lathe can be used to reduce the diameter of a part to a desired dimension. First, *clamp the part securely in a lathe chuck (636kB)*. The part should not extend more that three times its diameter. Then install a roughing or finishing tool (whichever is appropriate). If you're feeding the saddle toward the headstock (as in the clip below) use a right-hand turning tool. Move the tool off the part by backing the carriage up with the carriage handwheel, then use the cross feed to set the desired depth of cut. In the clip below, a finish cut is made using the power feed for a smoother finish. Remember that for each thousandth depth of cut, the work diameter is reduced by *two* thousandths. *(click here for video) 821kB*

Facing

A lathe can be used to create a smooth, flat, face very accurately perpendicular to the axis of a cylindrical part. First, *clamp the part securely in a lathe chuck (636kB)*. Then, install a facing tool. Bring the tool approximately into position, but slightly off of the part. Always *turn the spindle by hand (248kB)* before turning it on. This ensures that no parts

interfere with the rotation of the spindle. Move the tool outside the part and adjust the saddle to take the desired depth of cut. Then, feed the tool across the face with the cross slide. The following clip shows a roughing cut being made; about 50 thousandths are being removed in one pass. *(click here for video) 2.35MB* If a finer finish is required, take just a few thousandths on the final cut and use the power feed. Be careful clearing the ribbon-like chips; They are very sharp. Do not clear the chips while the spindle is turning. After facing, there is a very sharp edge on the part. *Break the edge (205kB)* with a file.

Parting

A parting tool is deeper and narrower than a turning tool. It is designed for making narrow grooves and for cutting off parts. When a parting tool is installed, ensure that it hangs over the tool holder enough that the the holder will clear the workpiece (but no more than that). Ensure that the parting tool is perpendicular to the axis of rotation and that the tip is the same height as the center of the part. A good way to do this is to hold the tool against the face of the part. Set the height of the tool, lay it flat against the face of the part, then lock the tool in place. *(click here for video) 2.45MB* When the cut is deep, the side of the part can rub against sides of the groove, so it's especially important to apply cutting fluid. In this clip, a part is cut off from a piece of stock. *(click here for video) 246kB*

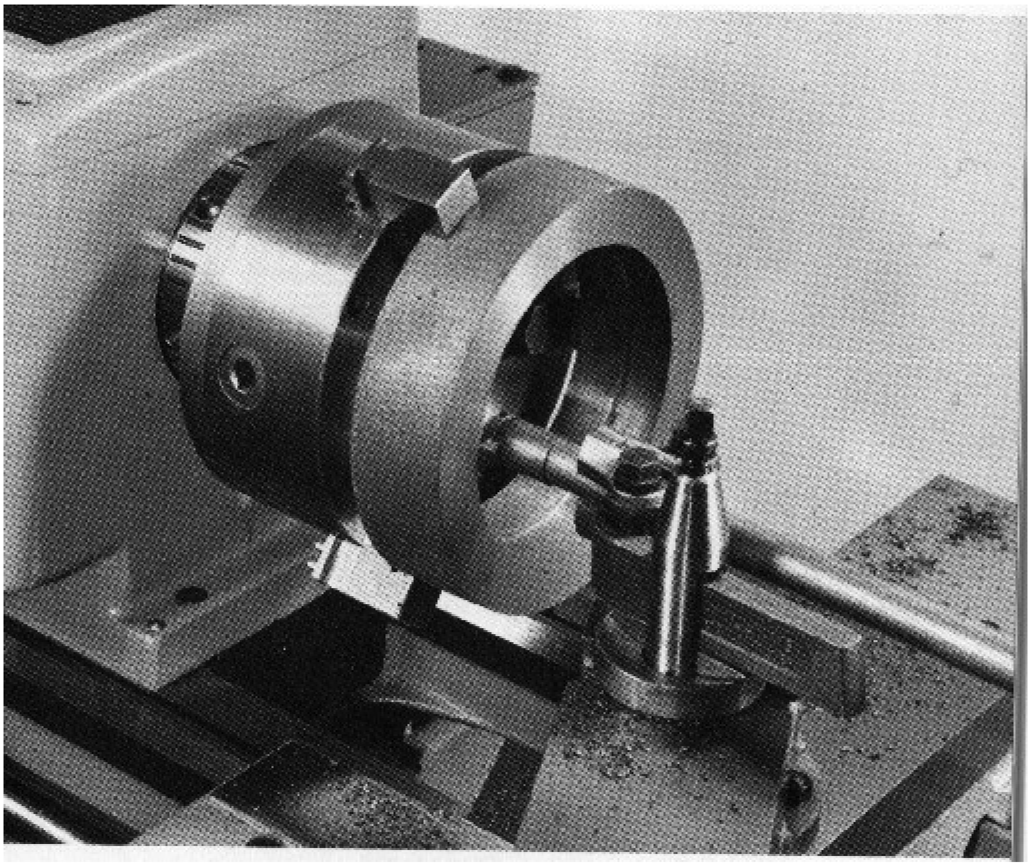
Drilling

A lathe can also be used to drill holes accurately concentric with the centerline of a cylindrical part. First, install a drill chuck into the tail stock. Make certain that the tang on the back of the drill chuck seats properly in the tail stock. Withdraw the jaws of the chuck and tap the chuck in place with a soft hammer. *(click here for video) 1.93MB*

Move the saddle forward to make room for the tailstock. Move the tailstock into position, and lock the it in place (otherwise it will slide backward as you try to drill). Before starting the machine, turn the spindle by hand. You've just moved the saddle forward, so it could interfere with the rotation of the lathe chuck. Always use a centerdrill to start the hole. *(click here for video) (2.30MB)*. You should use cutting fluid with the centerdrill. It has shallow flutes (for added stiffness) and doesn't cut as easily as a drill bit. Always drill past the beginning of the taper to create a funnel to guide the bit in. *(click for computer generated animation of funnel effect, not yet available)*. In this clip, a hole is drilled with a drill bit. *(click here for video) 1.07MB* Take at most one or two drill diameters of material before backing off, clearing the chips, and applying cutting fluid. If the drill bit squeeks, aplly solvent more often. The drill chuck can be removed from the tail stock by drawing back the drill chuck as far as it will easily go, then about a quarter turn more. A pin will press the chuck out of the collet. *(click here for video) 554kB*

Boring

Boring is an operation in which a hole is enlarged with a single point cutting tool. A boring bar is used to support the cutting tool as it extends into the hole. Because of the extension of the boring bar, the tool is supported less rigidly and is more likely to chatter. This can be corrected by using slower spindle speeds or by grinding a smaller radius on the nose of the tool.



Boring On a Lathe

Single Point Thread Turning

External threads can be cut with a die and internal threads can be cut with a tap. But for some diameters, no die or tap is available. In these cases, threads can be cut on a lathe. A special cutting tool should be used, typically witha 60 degree nose angle. To form threads with a specified number of threads per inch, the spindle is mechanically coupled to the carriage lead screw. Procedures vary for different machines.

Advanced Work Holding

Some parts require special techniques to hold them properly for lathe work. For instance, if you wish to cut on the entire outside diameter of a part, then the part cannot be held in a chuck or collet. If the part has a hole through it, you can press it on to a lathe arbor (a slightly tapered shaft), and clamp onto the arbor rather than the part itself. The hole must

have an adequate aspect ratio or the part will not be firmly supported. *(click here for video) 554kB*

If the part has a very large hole through it, a lathe arbor may not be a practicable solution. You may instead use the outside of the jaws to hold the inside diameter of the part. *(click here for video) 983kB*

If the part has a very complex geometry, it may be neccesary to install the part onto a face plate. The face plate is then attached to the spindle. *(click here for video) 452kB*

On to the **belt sander**.
Return to the **machine shop**.
Return to the **Mechanical Engineering Department**.

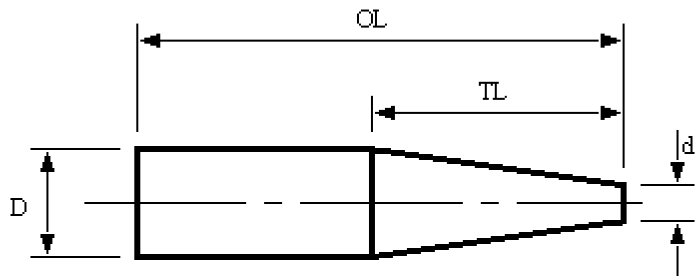
Maintained by ***2.670adm@mit.edu***

8.3.3 Turning Tapers on Lathes

· There are some common methods for turning tapers on a lathe,

1. - Off-setting the tail stock
2. - Using the compound slide
3. - using a taper turning attachment
4. - using a form tool

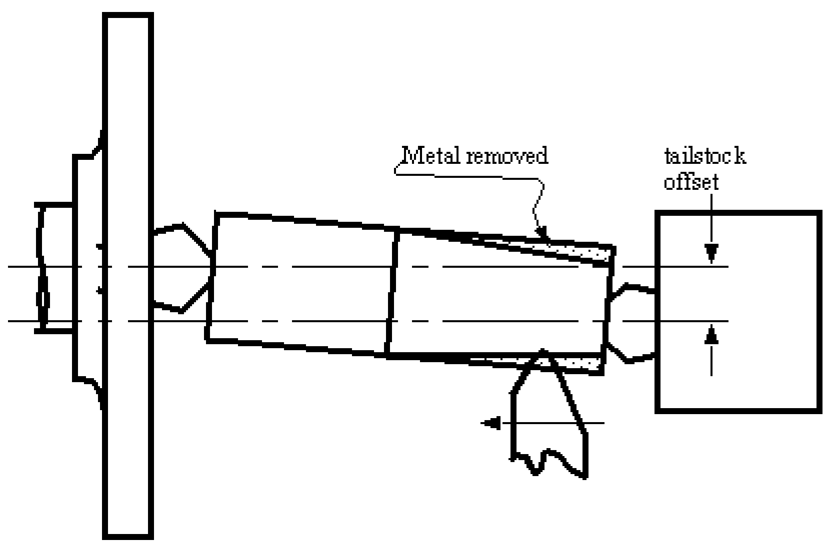
· Off-Set Tail Stock - In this method the normal rotating part of the lathe still drives the workpiece (mounted between centres), but the centre at the tailstock is offset towards/away from the cutting tool. Then, as the cutting tool passes over, the part is cut in a conical shape. The method for determining the offset distance is described below.



$$OFFSET = \frac{OL}{TL} \times \frac{(D-d)}{2} = \frac{tpf \times OL}{24}$$

where,

OL = overall length
TL = taper length
D = the large taper diameter
d = the small taper diameter
tpf = taper per foot (in.)
OFFSET = the distance to move the tailstock from the zero setting



It is necessary to measure the tailstock offset when using this method. This can be done with,

1. A scale
2. A dial indicator

This method is limited to small tapers over long lengths.

The misalignment of the centres used in this method can cause damage to the work, and to the centres.

· The Compound Slide Method - The compound slide is set to travel at half of the taper angle. The tool is then fed across the work by hand, cutting the taper as it goes.

· Taper Turning Attachment - Additional equipment is attached at the rear of the lathe. The cross slide is disconnected from the cross feed nut. The cross slide is then connected to the attachment. As the carriage is engaged, and travels along the bed, the attachment will cause the cutter to move in/out to cut the taper.

· Form Tool - This type of tool is specifically designed for one cut, at a certain taper angle. The tool is plunged at one location, and never moved along the lathe slides.

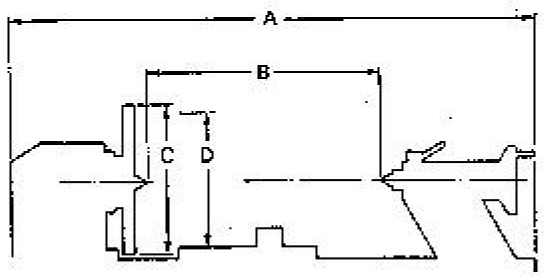


Fig. 10-3. Study how the lathe is measured. A-Length of bed. B-Distance between centers. C-Diameter of work that can be turned over two ways. D-Diameter of work that can be turned over cross slide.

LATHE SAFETY

WARNING! Lathe chips are sharp; do NOT remove them with your hands.
DANGER! an air hose should NEVER be used to remove chips. The flying particles might injure you or a nearby person.

1. No attempt should be made to operate a lathe until you know the proper procedures and have been checked out on its safe operation by your instructor.
2. Dress appropriately! Remove necktie, necklace, wrist watch, rings and other jewelry, and loose fitting sweaters. Wear an apron or a properly fitted shop coat. Safety glasses are a must!
3. Clamp all work solidly! Use the correct size tool and work holding device for the job. Get help when handling large sections of metal and heavy chucks and attachments.

4. Check work frequently when it is being machined between centers. The work expands as it heats up and could damage the tailstock center.
5. Be sure all guards are in place before attempting to operate the machine.
6. Turn the faceplate or chuck by hand to be sure there is NO binding or danger of the work striking any part of the lathe.
7. Keep the machine clear of tools!
8. Stop the machine before making measurements and adjustments.
9. Remember--chips are sharp! Do NOT try to remove them with your hands when they become "stringy" and build up on the tool post. Stop the machine and remove them with pliers.
10. Do NOT permit small diameter work to project too far from the chuck without support from the tailstock. Without support, the work will be tapered, or worse, spring up over the cutting tool and/or break.
11. Be careful NOT to run the cutting tool into the chuck. Check any readjustment of work or tool for ample clearance when the cutter has been moved left to the farthest point that will be machined.
12. Stop the machine before attempting to wipe down, a machine surface.
13. Before repositioning or removing work from the lathe, move the cutting tool clear of the work area. This will prevent accidental cuts from the cutter bit.
14. Avoid talking to anyone while running a lathe! Do NOT permit anyone to fool around with the machine while you are operating it. You are the only one who should turn the machine on or off, or make adjustments to the lathe.
15. If the lathe has a threaded spindle nose, never attempt to run the chuck on or off the spindle using power. It is also dangerous practice to stop such a lathe by reversing the direction of rotation. The chuck could spin off and cause serious injury to you. There is also the danger of damaging the machine.
16. You should always be aware of the direction of travel and speed of the carriage before engaging the half-nuts or automatic feed.
17. Always remove the key from the chuck. Make it a habit NEVER to let go of the key until it is out of the chuck and clear of the work area.
18. Tools must NOT be placed on the lathe ways. Use a tool board or place them on the lathe tray.
19. When filing on the lathe, be sure the file has a securely fitting handle.
20. Stop the machine immediately if some off sounding noise or vibration develops during operation. If you cannot locate the trouble, get help from your instructor. Under no condition should the machine be operated until the trouble has been corrected.
21. Remove sharp edges and burrs from work before removing it from the machine.
22. Plan your work thoroughly before starting. Have all needed tools on hand.

23. Use care when cleaning the lathe. Chips sometimes stick in recesses. Remove them with a brush or short stick., NEVER clean a machine tool with compressed air.

DANGER! Stop the machine before making measurements or cleaning out chips!

SAFETY NOTE! Under NO condition should a lathe be reversed to brake it to a stop!

INTRODUCTION

What is turning?

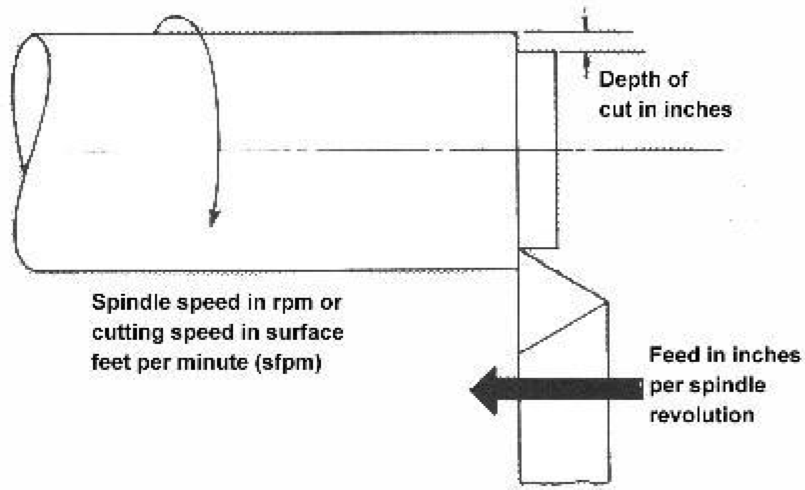
Turning is the machining operation that produces cylindrical parts. In its basic form, it can be defined as the machining of an external surface:

≅ with the workpiece rotating,
≅ with a single-point cutting tool, and
≅ with the cutting tool feeding parallel to the axis of the workpiece and at a distance that will remove the outer surface of the work.

Taper turning is practically the same, except that the cutter path is at an angle to the work axis. Similarly, in contour turning, the distance of the cutter from the work axis is varied to produce the desired shape.

Even though a single-point tool is specified, this does not exclude multiple-tool setups, which are often employed in turning. In such setups, each tool operates independently as a single-point cutter.

View a typical turning operation. This movie is from the MIT-NMIS Machine Shop Tutorial.



Turning and the adjustable parameters

Adjustable cutting factors in turning

The three primary factors in any basic turning operation are speed, feed, and depth of cut. Other factors such as kind of material and type of tool have a large influence, of course, but these three are the ones the operator can change by adjusting the controls, right at the machine.

Speed, always refers to the spindle and the workpiece. When it is stated in revolutions per minute(rpm) it tells their rotating speed. But the important figure for a particular turning operation is the surface speed, or the speed at which the workpeece material is moving past the cutting tool. It is simply the product of the rotating speed times the circumference (in feet) of the workpiece before the cut is started. It is expressed in surface feet per minute (sfpm), and it refers only to the workpiece. Every different diameter on a workpiece will have a different cutting speed, even though the rotating speed remains the same.

Feed, always refers to the cutting tool, and it is the rate at which the tool advances along its cutting path. On most power-fed lathes, the feed rate is directly related to the spindle speed and is expressed in inches (of tool advance) per revolution (of the spindle), or ipr. The figure, by the way, is usually much less than an inch and is shown as decimal amount.

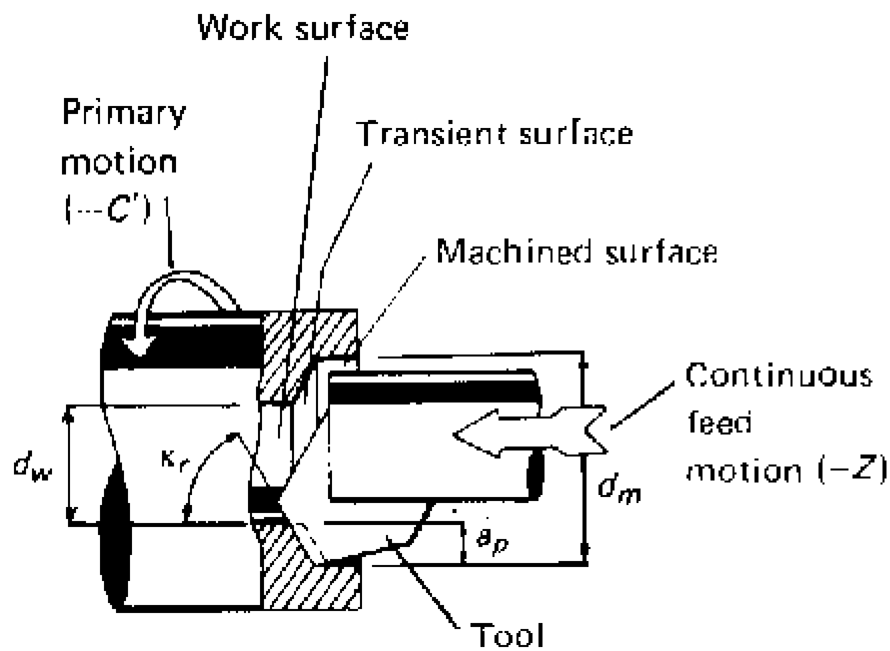
Depth of Cut, is practically self explanatory. It is the thickness of the layer being removed from the workpiece or the distance from the uncut surface of the work to the cut surface, expressed in inches. It is important to note, though, that the diameter of the workpiece is reduced by two times the depth of cut because this layer is being removed from both sides of the work.

Return to Top

LATHE RELATED OPERATIONS

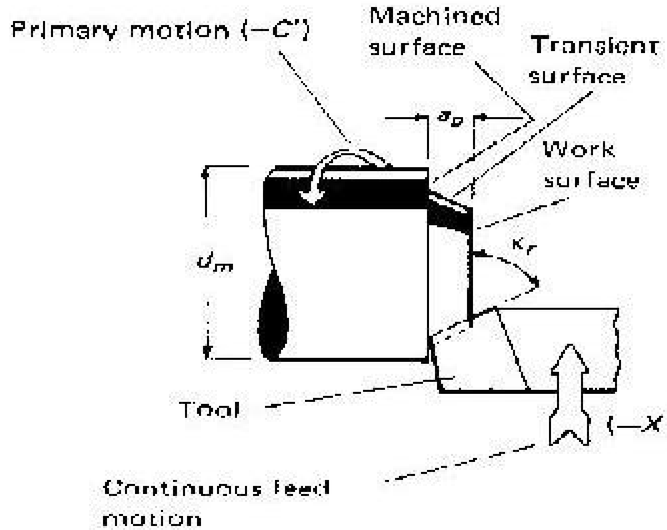
The lathe, of course, is the basic turning machine. Apart from turning, several other operations can also be performed on a lathe.

Boring. Boring always involves the enlarging of an existing hole, which may have been made by a drill or may be the result of a core in a casting. An equally important, and concurrent, purpose of boring may be to make the hole concentric with the axis of rotation of the workpiece and thus correct any eccentricity that may have resulted from the drill's having drifted off the center line. Concentricity is an important attribute of bored holes. When boring is done in a lathe, the work usually is held in a chuck or on a face plate. Holes may be bored straight, tapered, or to irregular contours. Boring is essentially internal turning while feeding the tool parallel to the rotation axis of the workpiece.

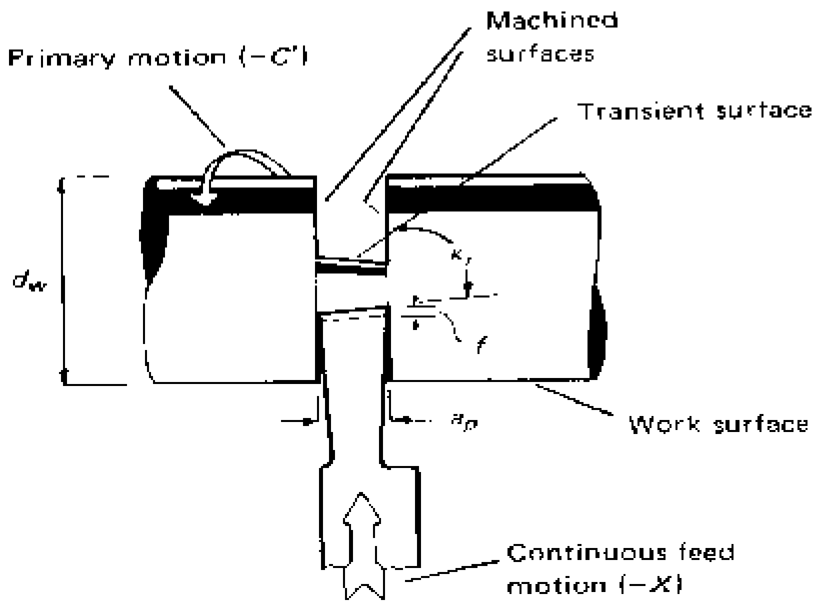


Facing. Facing is the producing of a flat surface as the result of a tool's being fed across the end of the rotating workpiece. Unless the work is held on a mandrel, if both ends of the work are to be faced, it must be turned end for end after the first end is completed and the facing operation repeated. The cutting speed should be determined from the largest diameter of the surface to be faced. Facing may be done either from the outside inward or from the center outward. In either case, the point of the tool must be set exactly at the height of the center of rotation. because the cutting force tends to push the tool away from the work, it is usually desirable to clamp the carriage to the lathe bed during each facing cut to prevent it from moving slightly and thus producing a surface that is not flat. In the facing of casting or other materials that have a hard surface, the depth of the first

cut should be sufficient to penetrate the hard material to avoid excessive tool wear.

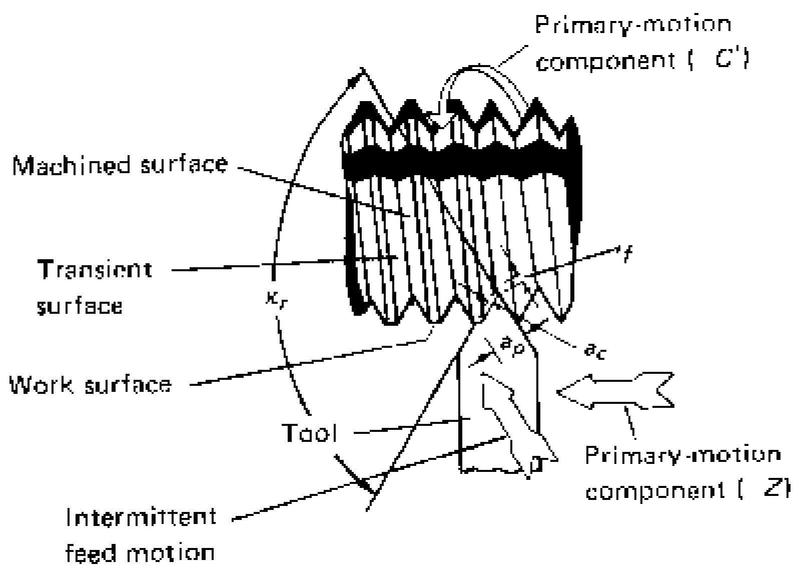


Parting. Parting is the operation by which one section of a workpiece is severed from the remainder by means of a cutoff tool. Because cutting tools are quite thin and must have considerable overhang, this process is less accurate and more difficult. The tool should be set exactly at the height of the axis of rotation, be kept sharp, have proper clearance angles, and be fed into the workpiece at a proper and uniform feed rate.



Threading. Lathe provided the first method for cutting threads by machines. Although most threads are now produced by other methods, lathes still provide the most versatile and fundamentally simple method. Consequently, they often are used for cutting threads on special workpieces where the configuration or nonstandard size does not permit them to be made by less costly methods. There are two basic requirements for thread cutting. An accurately shaped and properly mounted tool is needed because thread cutting is a

form-cutting operation. The resulting thread profile is determined by the shape of the tool and its position relative to the workpiece. The second by requirement is that the tool must move longitudinally in a specific relationship to the rotation of the workpiece, because this determines the lead of the thread. This requirement is met through the use of the lead screw and the split unit, which provide positive motion of the carriage relative to the rotation of the spindle.

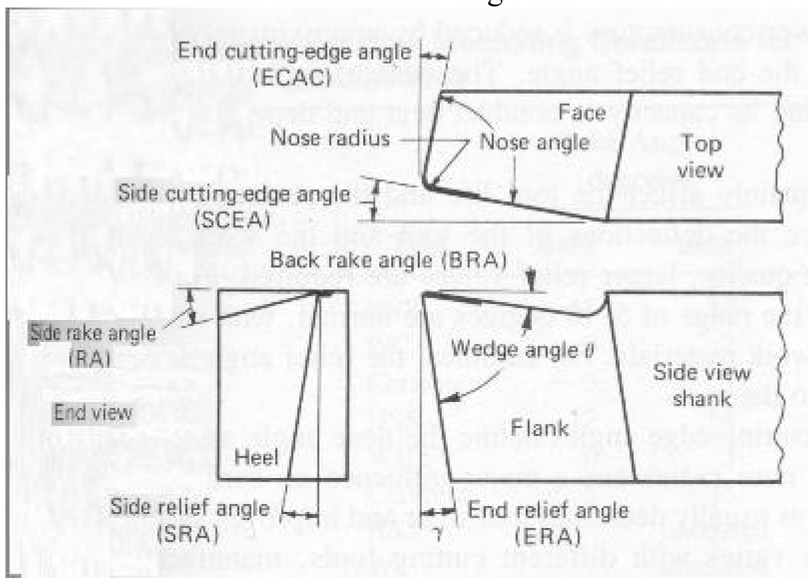


Return to Top

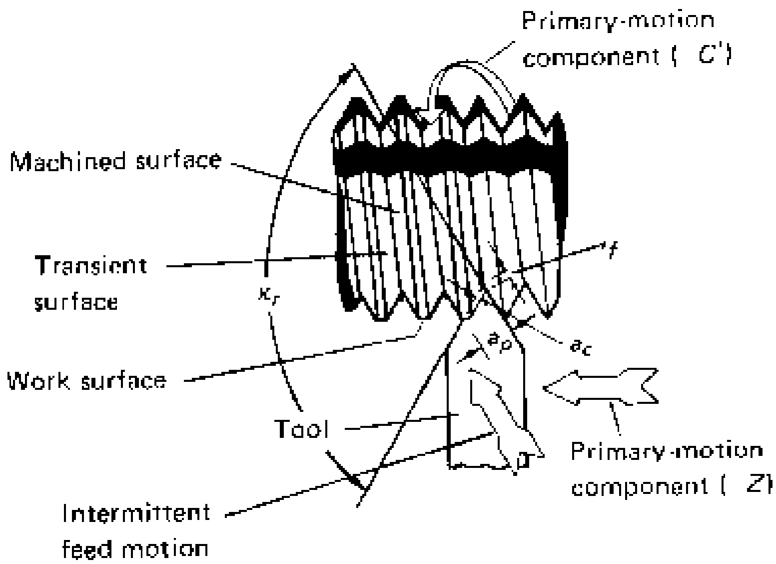
CUTTING TOOLS FOR LATHES

Tool Geometry. For cutting tools, geometry depends mainly on the properties of the tool material and the work material. The standard terminology is shown in the following figure. For single point tools, the most important angles are the rake angles and the end

and side relief angles.



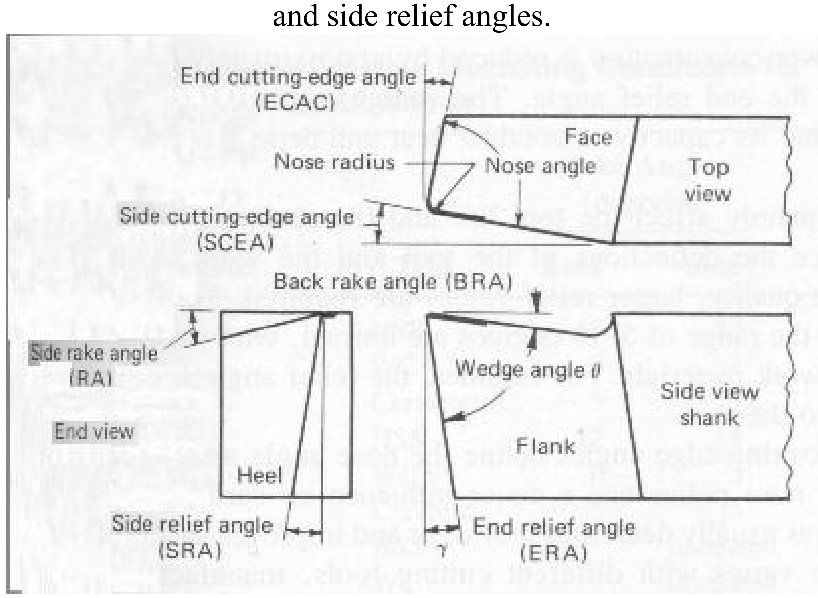
The back rake angle affects the ability of the tool to shear the work material and form the chip. It can be positive or negative. Positive rake angles reduce the cutting forces resulting in smaller deflections of the workpiece, tool holder, and machine. If the back rake angle is too large, the strength of the tool is reduced as well as its capacity to conduct heat. In machining hard work materials, the back rake angle must be small, even negative for carbide and diamond tools. The higher the hardness, the smaller the back rake angle. For high-speed steels, back rake angle is normally chosen in the positive range. There are two basic requirements for thread cutting. An accurately shaped and properly mounted tool is needed because thread cutting is a form-cutting operation. The resulting thread profile is determined by the shape of the tool and its position relative to the workpiece. The second by requirement is that the tool must move longitudinally in a specific relationship to the rotation of the workpiece, because this determines the lead of the thread. This requirement is met through the use of the lead screw and the split unit, which provide positive motion of the carriage relative to the rotation of the spindle.



Return to Top

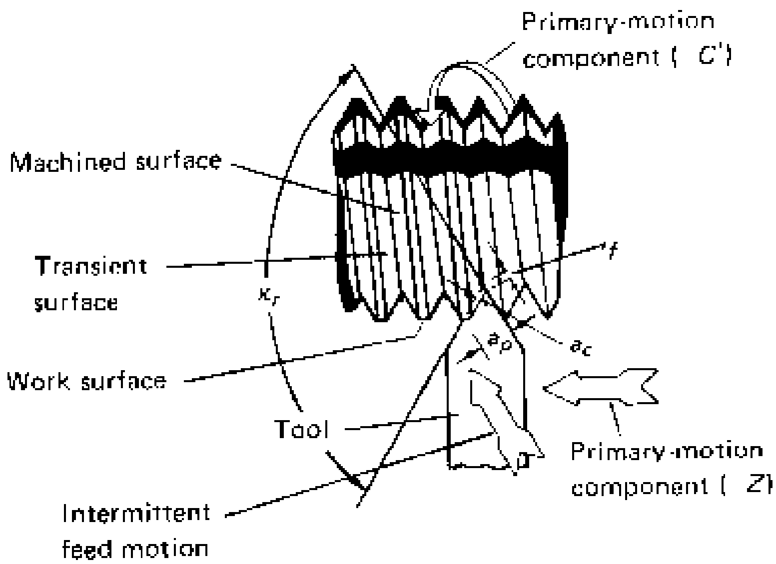
CUTTING TOOLS FOR LATHES

Tool Geometry. For cutting tools, geometry depends mainly on the properties of the tool material and the work material. The standard terminology is shown in the following figure. For single point tools, the most important angles are the rake angles and the end and side relief angles.



The back rake angle affects the ability of the tool to shear the work material and form the chip. It can be positive or negative. Positive rake angles reduce the cutting forces resulting in smaller deflections of the workpiece, tool holder, and machine. If the back

rake angle is too large, the strength of the tool is reduced as well as its capacity to conduct heat. In machining hard work materials, the back rake angle must be small, even negative for carbide and diamond tools. The higher the hardness, the smaller the back rake angle. For high-speed steels, back rake angle is normally chosen in the positive range. There are two basic requirements for thread cutting. An accurately shaped and properly mounted tool is needed because thread cutting is a form-cutting operation. The resulting thread profile is determined by the shape of the tool and its position relative to the workpiece. The second by requirement is that the tool must move longitudinally in a specific relationship to the rotation of the workpiece, because this determines the lead of the thread. This requirement is met through the use of the lead screw and the split unit, which provide positive motion of the carriage relative to the rotation of the spindle.

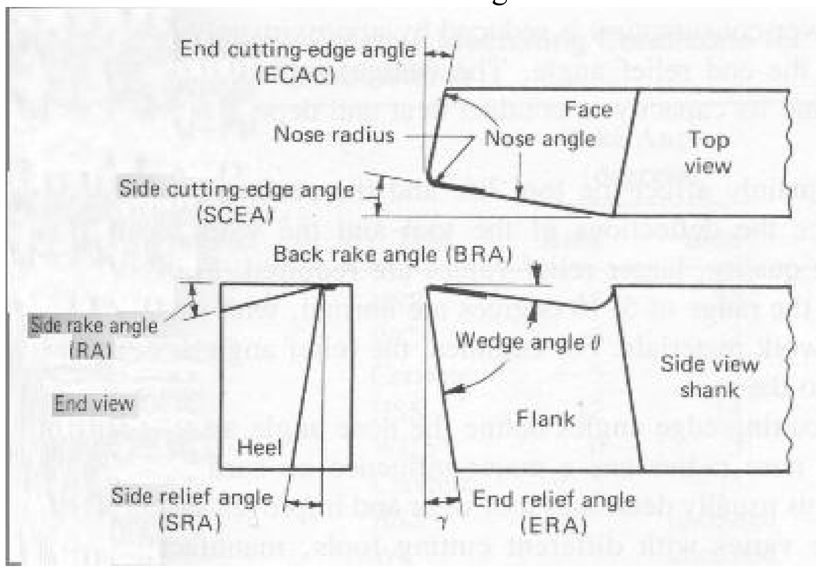


Return to Top

CUTTING TOOLS FOR LATHES

Tool Geometry. For cutting tools, geometry depends mainly on the properties of the tool material and the work material. The standard terminology is shown in the following figure. For single point tools, the most important angles are the rake angles and the end

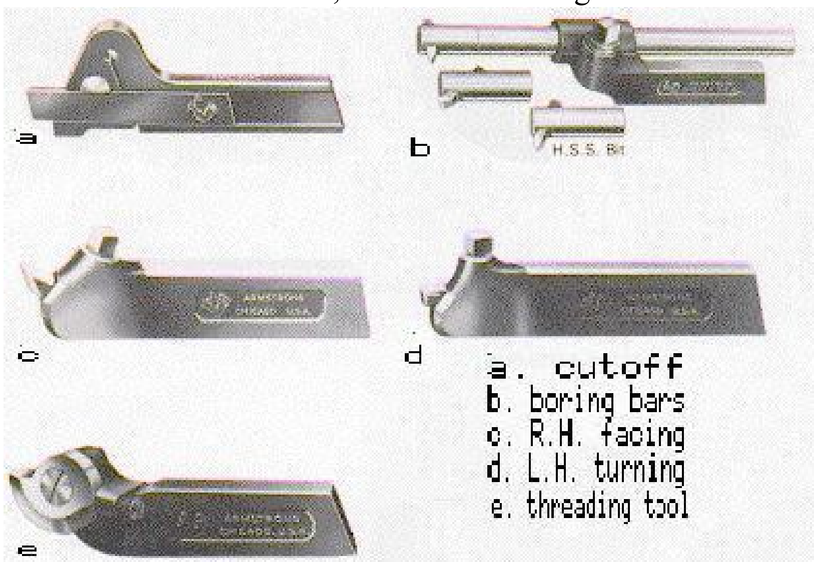
and side relief angles.



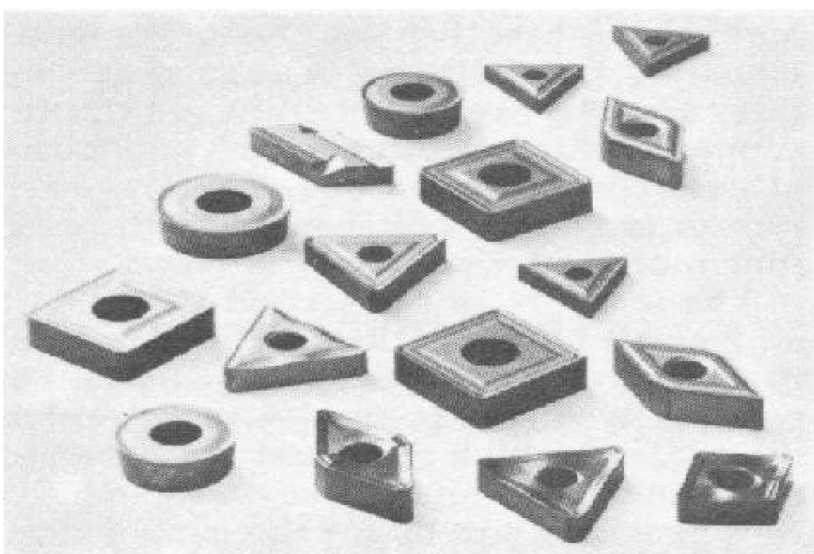
The back rake angle affects the ability of the tool to shear the work material and form the chip. It can be positive or negative. Positive rake angles reduce the cutting forces resulting in smaller deflections of the workpiece, tool holder, and machine. If the back rake angle is too large, the strength of the tool is reduced as well as its capacity to conduct heat. In machining hard work materials, the back rake angle must be small, even negative for carbide and diamond tools. The higher the hardness, the smaller the back rake angle. For high-speed steels, back rake angle is normally chosen in the positive range.

Most lathe operations are done with relatively simple, single-point cutting tools. On right-hand and left-hand turning and facing tools, the cutting takes place on the side of the tool; therefore the side rake angle is of primary importance and deep cuts can be made. On the round-nose turning tools, cutoff tools, finishing tools, and some threading tools, cutting takes place on or near the end of the tool, and the back rake is therefore of importance. Such tools are used with relatively light depths of cut. Because tool materials are expensive, it is desirable to use as little as possible. It is essential, at the same, that the cutting tool be supported in a strong, rigid manner to minimize deflection and possible vibration. Consequently, lathe tools are supported in various types of heavy, forged steel

tool holders, as shown in the figure.

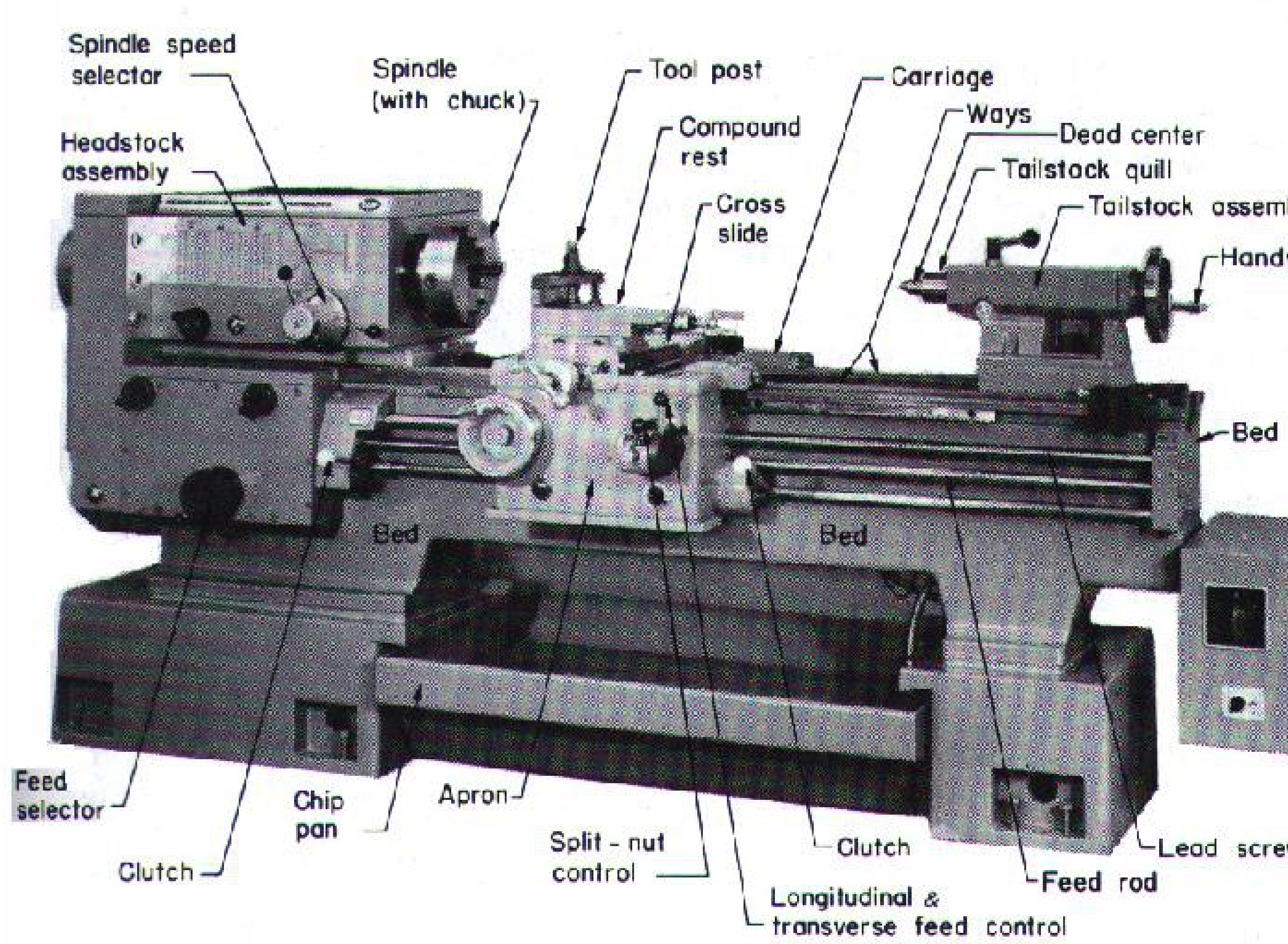


The tool bit should be clamped in the tool holder with minimum overhang. Otherwise, tool chatter and a poor surface finish may result. In the use of carbide, ceramic, or coated carbides for mass production work, throwaway inserts are used; these can be purchased in great variety of shapes, geometrics (nose radius, tool angle, and groove geometry), and sizes.



TURNING MACHINES

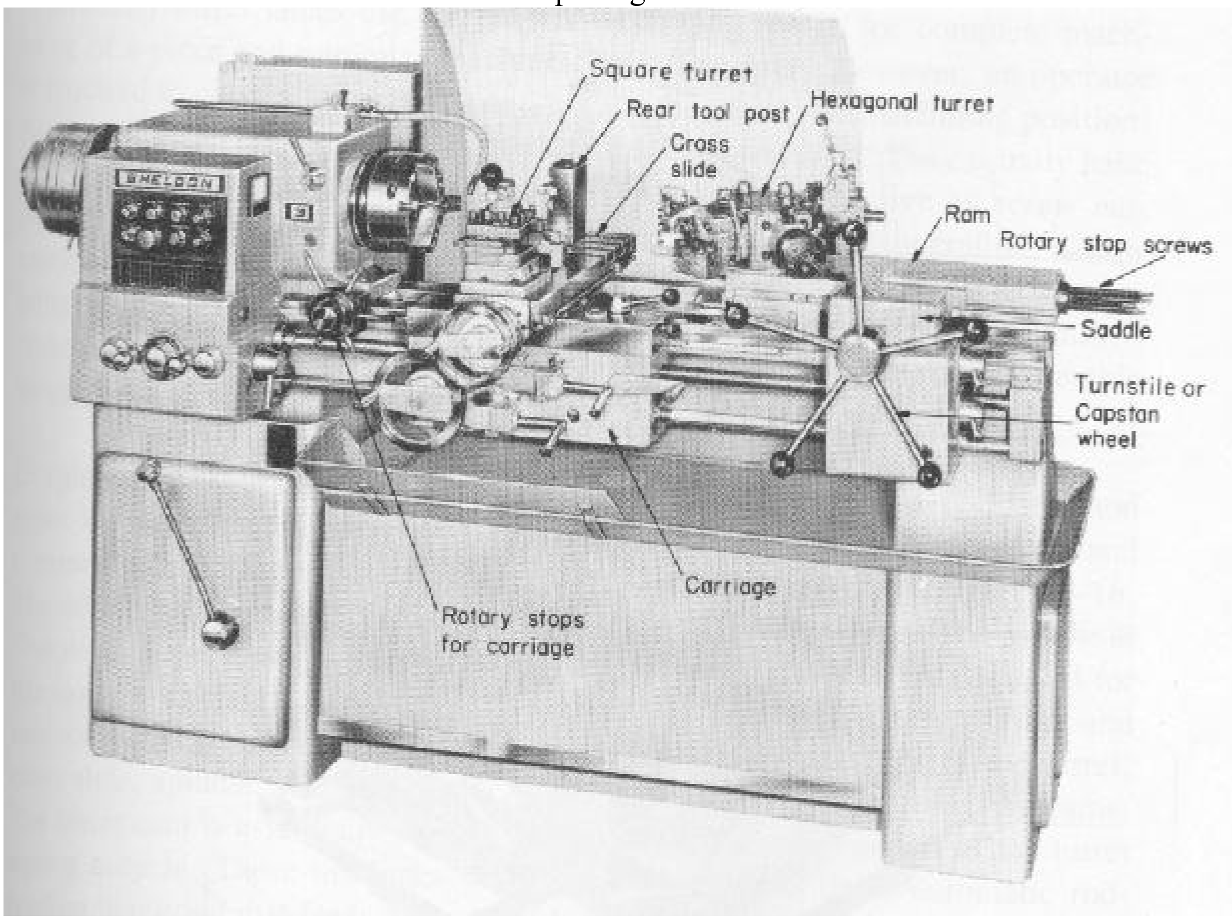
The turning machines are, of course, every kinds of lathes. Lathes used in manufacturing can be classified as engine, turret, automatics, and numerical control etc.



They are heavy duty machine tools and have power drive for all tool movements. They commonly range in size from 12 to 24 inches swing and from 24 to 48 inches center distance, but swings up to 50 inches and center distances up to 12 feet are not uncommon. Many engine lathes are equipped with chip pans and built-in coolant circulating system.

Turret Lathes. In a turret lathe, a longitudinally feedable, hexagon turret replaces the tailstock. The turret, on which six tools can be mounted, can be rotated about a vertical axis to bring each tool into operating position, and the entire unit can be moved longitudinally, either annually or by power, to provide feed for the tools. When the turret assembly is backed away from the spindle by means of a capstan wheel, the turret indexes automatically at the end of its movement thus bringing each of the six tools into operating position. The square turret on the cross slide can be rotated manually about a vertical axis to bring each of the four tools into operating position. On most machines, the turret can be moved transversely, either manually or by power, by means of the cross

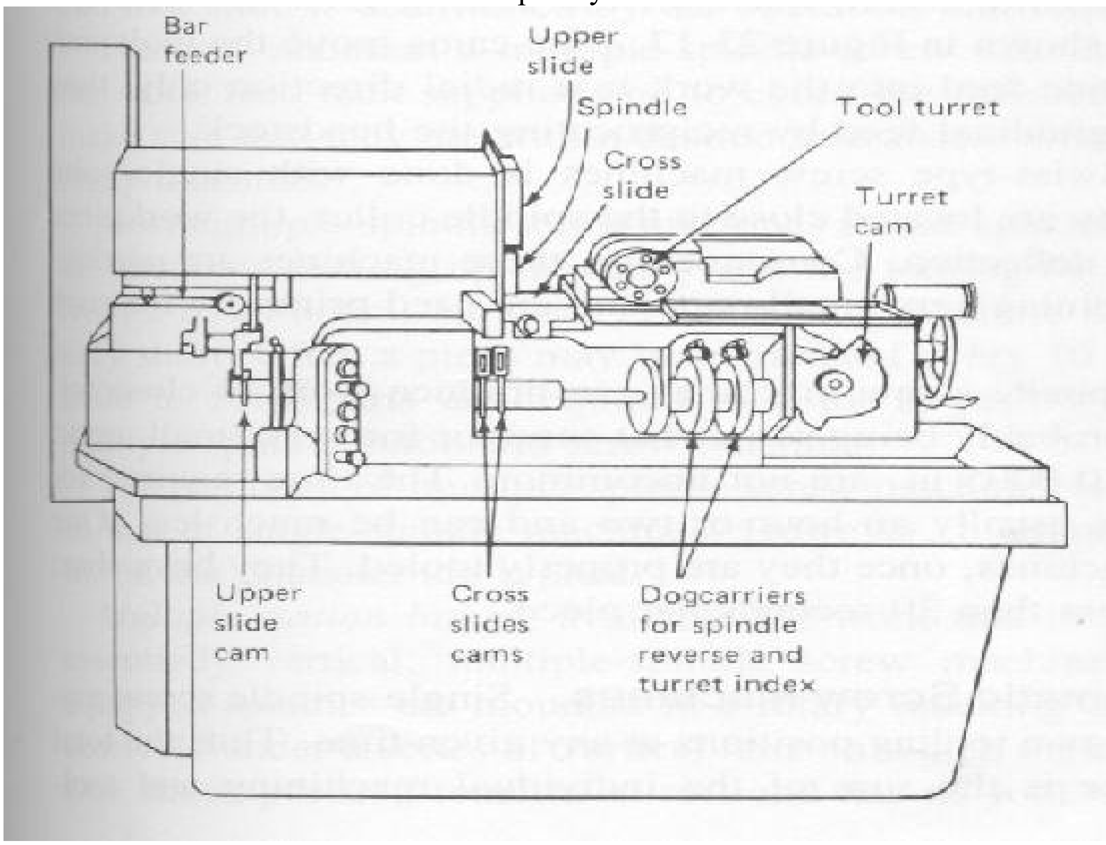
slide, and longitudinally through power or manual operation of the carriage. In most cased, a fixed tool holder also is added to the back end of the cross slide; this often carries a parting tool.



Through these basic features of a turret lathe, a number of tools can be set on the machine and then quickly be brought successively into working position so that a complete part can be machined without the necessity for further adjusting, changing tools, or making measurements.

Single-Spindle Automatic Screw Machines. There are two common types of single-spindle screw machines, One, an American development and commonly called the turret type (Brown & Sharp), is shown in the following figure. The other is of Swiss origin and is referred to as the swiss type. The Brown & Sharp screw machine is essentially a small automatic turret lathe, designed for bar stock, with the main turret mounted on the cross slide. All motions of the turret, cross slide, spindle, chuck, and stock-feed mechanism are controlled by cams. The turret cam is essentially a program that defines the movement of the turret during a cycle. These machines usually are equipped with an automatic rod feeding magazine that feeds a new length of bar stock into the collect as soon as one rod

is completely used.



CNC Machines. Nowadays, more and more Computer Numerical Controlled (CNC) machines are being used in every kinds of manufacturing processes. In a CNC machine, functions like program storage, tool offset and tool compensation, program-editing capability, various degree of computation, and the ability to send and receive data from a variety of sources, including remote locations can be easily realized through on board computer. The computer can store multiple-part programs, recalling them as needed for different parts. A CNC turret lathe in Michigan Technological University is shown in the following picture.

MILLING

Introduction

Milling Equipment

Milling Process Performance*(Software available here)*

Milling Research

Milling Technology Update

Go to the MANUFACTURING EDUCATION PAGE

Comments or suggestions: Contact Prof. John. W. Sutherland at jwsuther@mtu.edu

INTRODUCTION

WHAT IS MILLING?

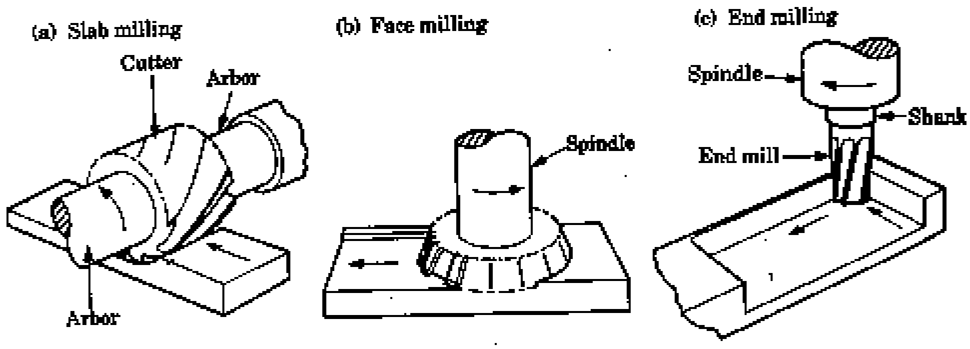
Milling is the process of cutting away material by feeding a workpiece past a rotating multiple tooth cutter. The cutting action of the many teeth around the milling cutter provides a fast method of machining. The machined surface may be flat,angular, or curved. The surface may also be milled to any combination of shapes. The machine for holding the workpiece, rotating the cutter, and feeding it is known as the Milling machine.

View a typical milling operation. This movie is from the MIT-NMIS Machine Shop Tutorial

CLASSIFICATION OF MILLING

ü Peripheral Milling

In peripheral (or slab) milling, the milled surface is generated by teeth located on the periphery of the cutter body. The axis of cutter rotation is generally in a plane parallel to the workpiece surface to be machined.



(Kalpakjian S., Introduction to Manufacturing Processes)

ü Face Milling

In face milling, the cutter is mounted on a spindle having an axis of rotation perpendicular to the workpiece surface. The milled surface results from the action of cutting edges located on the periphery and face of the cutter.

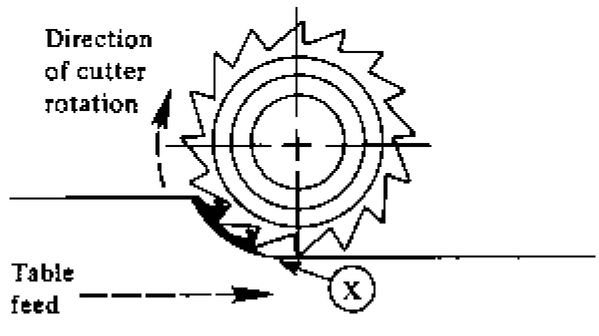
ü End Milling

The cutter in end milling generally rotates on an axis vertical to the workpiece. It can be tilted to machine tapered surfaces. Cutting teeth are located on both the end face of the cutter and the periphery of the cutter body.

METHODS OF MILLING

- ***Up Milling***

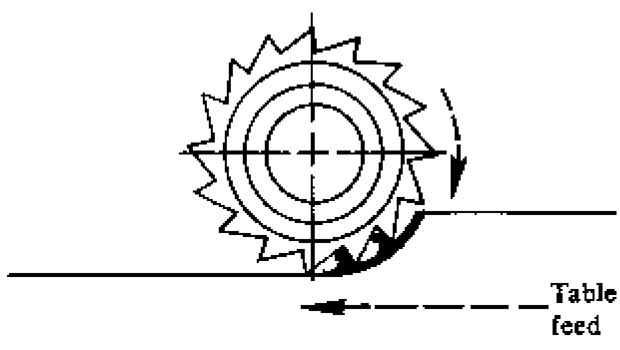
Up milling is also referred to as conventional milling. The direction of the cutter rotation opposes the feed motion. For example, if the cutter rotates clockwise , the workpiece is fed to the right in up milling.



(Boothroyd G. & Knight W., Fundamentals of Machining and Machine Tools)

- ***Down Milling***

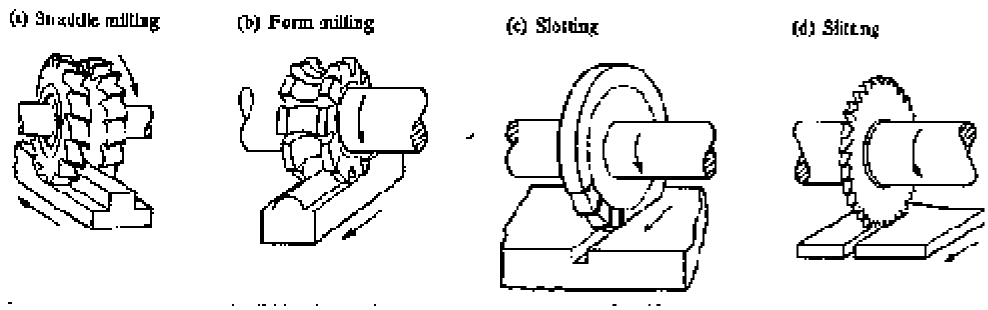
Down milling is also referred to as climb milling. The direction of cutter rotation is same as the feed motion. For example, if the cutter rotates counterclockwise , the workpiece is fed to the right in down milling.



(Boothroyd G. & Knight W., Fundamentals of Machining and Machine Tools)

The chip formation in down milling is opposite to the chip formation in up milling. The figure for down milling shows that the cutter tooth is almost parallel to the top surface of the workpiece. The cutter tooth begins to mill the full chip thickness. Then the chip thickness gradually decreases.

Other milling operations are shown in the figure.



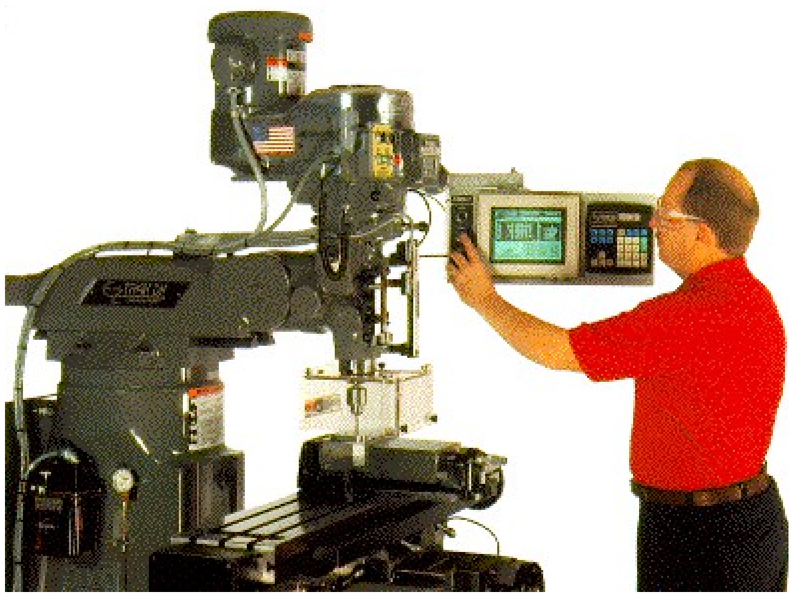
(Kalpakjian S., Introduction to Manufacturing Processes)

Go to the MILLING PAGE

Go to the MANUFACTURING EDUCATION PAGE

MILLING EQUIPMENT

The milling machine is one of the most versatile machine tools in existence. In addition to straight milling of flat and irregularly shaped surfaces, it can perform gear and thread cutting, drilling, boring and slotting operations which are normally handled on machine tools designed specifically for these specific operations.



The above is a Bridgeport CNC Milling Machine

Types of Milling Machines

Milling machines can be broadly classified into the following types:

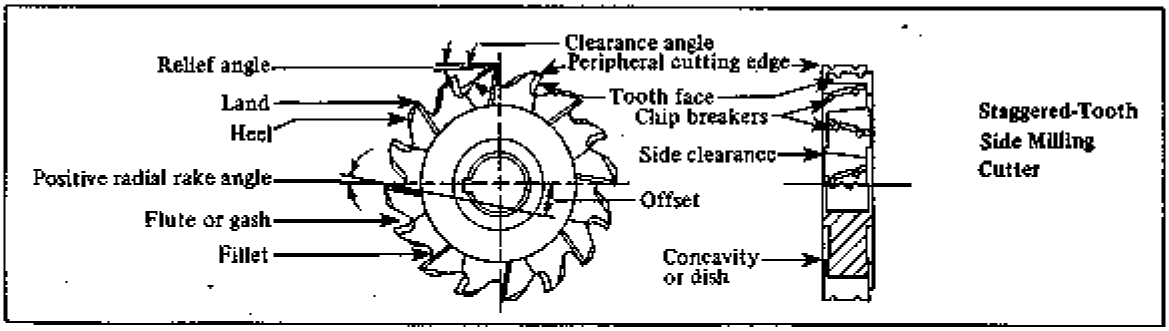
- ***Column and knee type of milling machines***
- ***Bed type***
- ***Rotary table***
- ***Tracer controlled***

Milling Cutters

A milling cutter is a cutting tool that is used on a milling machine. Milling cutters are available in many standard and special types, forms, diameters, and widths.The teeth maybe straight (parallel to the axis of rotation) or at a helix angle. The helix angle helps a slow engagement of the tool distributing the forces .The cutter may be right-hand (to turn clockwise) or left-hand (to turn counterclockwise).The figure shows a typical end milling cutter.

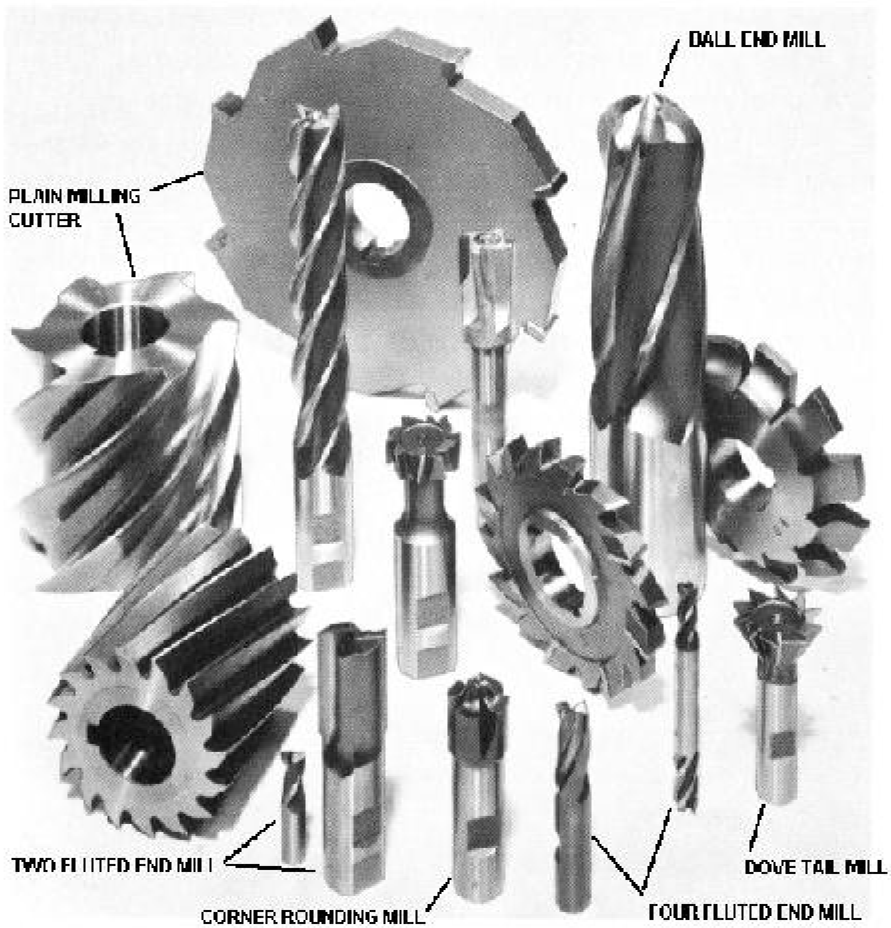
Features of Milling Cutters

Some of the terms used to identify the major features of a milling cutter are given in the figure.



(Olivo C.T., Machine Tool Technology and Manufacturing Processes, C Thomas Olivo and Associates)

Types of Milling Cutters


BALL END MILL
PLAIN MILLING
CUTTER
TWO FLUTED END MILL
CORNER ROUNDING MILL
FOUR FLUTED END MILL
DOVE TAIL MILL

Surface grinder

From Wikipedia, the free encyclopedia
Jump to: navigation, search

A **surface grinder** is a machine tool used to provide precision ground surfaces, either to a critical size or for the surface finish.

Surface Grinder with electromagnetic chuck, inset shows a Manual magnetic chuck

The typical precision of a surface grinder depends on the type and usage, however +/- 0.002 mm (+/- 0.0001") should be achievable on most surface grinders.

The machine consists of a table that traverses both longitudinally and across the face of the wheel. The longitudinal feed is usually powered by hydraulics, as may the cross feed, however any mixture of hand, electrical or hydraulic may be used depending on the ultimate usage of the machine (ie: production, workshop, cost). The grinding wheel rotates in the spindle head and is also adjustable for height, by any of the methods described previously. Modern surface grinders are semi-automated, depth of cut and spark-out may be preset as to the number of passes and once setup the machining process requires very little operator intervention.

Spark out is a term used when precision values are sought and literally means "until the sparks are out (no more)". It involves passing the workpiece under the wheel, without resetting the depth of cut, more than once and generally multiple times. This ensures that any inconsistencies in the machine or workpiece are eliminated.

As with any grinding operation, the condition of the wheel is extremely important. Diamond dressers are used to maintain the condition of the wheel, these may be table mounted or as the first image shows, mounted in the wheel head where they can be readily applied.

The machine has provision for the application of coolant as well as the extraction of metal dust (metal and grinding particles).

Depending on the workpiece material, the work is generally held by the use of a magnetic chuck. This may be either an electromagnetic chuck or a manually operated, both types are shown in the first image.